REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*
30/11/2009

2. REPORT TYPE
Final Report

3. DATES COVERED *(From - To)*
3/1/2006 - 8/31/2009

4. TITLE AND SUBTITLE
Moving Geometries and Viscous Flows Using Embedded-boundary Cartesian Grids

5a. CONTRACT NUMBER
FA9550-06-1-0203

5b. GRANT NUMBER
NA

5c. PROGRAM ELEMENT NUMBER
NA

6. AUTHOR(S)
Marsha Berger

5d. PROJECT NUMBER
NA

5e. TASK NUMBER
NA

5f. WORK UNIT NUMBER
NA

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Courant Institute of
Mathematical Sciences, New
York University
251 Mercer Street
New York, NY 10012-1019

8. PERFORMING ORGANIZATION REPORT NUMBER
25-74100-F5301

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
USAF, AFRL
AF Office of Scientific Research
875 North Randolph Street, Room 3112
Arlington, VA 22203

10. SPONSOR/MONITOR'S ACRONYM(S)
AFOSR

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
Code FA9550

12. DISTRIBUTION / AVAILABILITY STATEMENT
Distribution Statement A: Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES:
The views and conclusions contained herein are those of the author and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the AFOSR or US Govt.

14. ABSTRACT

We describe steps taken towards the development of an embedded boundary method for discretizing the Navier Stokes equations in complex geometry. New discretizations were developed and tested for use at cut cells and at mesh interfaces. A multigrid mesh coarsening algorithm was developed to accelerate convergence of the solution to steady state. Comparisons with several standard test cases from the literature were performed. This is preliminary work in two space dimensions towards our goal of extending Cart3D to be able to simulate three-dimensional viscous flows.

15. SUBJECT TERMS
embedded boundary grids, cut cells, viscous flow

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
					Marsha Berger
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**	Unlimited	5	**19b. TELEPHONE NUMBER** *(include area code)*
U	U	U			212-998-3305

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18

MOVING GEOMETRIES AND VISCOUS FLOW USING EMBEDDED-BOUNDARY CARTESIAN GRIDS

FINAL REPORT

AFOSR FA9550-06-1-0203

Marsha Berger
Courant Institute
New York University

Executive Summary

Our long-term goal is the development of automated tools for computing fluid flow in complicated geometries. In the last decade, we have developed both algorithms and software for mesh generation and adaptation, surface preparation, multi-grid coarse grid generation, and a highly parallel steady-state inviscid flow solver. In combination with other tools (in particular the ability to generate high quality triangulations directly from CAD using CAPRI, developed by Bob Haimes and Michael Aftosmis), complete simulations can be performed with greatly reduced set up time. In the last two years a significant advance is the development of an adjoint module for Cart3D by Aftosmis and Nemec. This is used to automatically drive the mesh adaptation, and produce an error estimate for an objection function such a lift or drag.

A specific goal of this project was to extend Cartesian embedded boundary methods to solve the compressible viscous Navier-Stokes equations. The first step in this direction was to develop a discretization for the viscous terms. This was done for two dimensional problems, and several test cases from the literature were studied. Future work will include extensions to three dimensions, turbulence modeling, and a numerical method or model for handling the boundary layer without resorting to the prohibitively expensive use of isotropic refinement. This work is in collaboration with Michael Aftosmis and Marian Nemec of NASA Ames.

Summary of Research

The Cart3D flow solver was extended to include viscous terms in two dimensions. The first step in modifying the steady-state inviscid code to simulate the Navier-Stokes equations was to develop discretizations for the second derivative terms at cut cells and at mesh interfaces. We examined four different stencils for use at cut cells in an elliptic model problem. We chose the best one to implement in the Navier Stokes code. We found it necessary to use a quadratic reconstruction in the normal direction at the cut cells, but we retained the linear gradient for tangential reconstruction. We also experimented with using a second order re-centering at the mesh interfaces in the direction normal to the interface. Doing this only in the one direction was not sufficient to reduce the overall error, and we are sticking with the linear reconstruction in these parts of the mesh. This development work was in two space dimensions.

We experimented with several test cases from the literature. First we ran experiments for a Blasius boundary layer profile with a non-coordinate aligned grid. As was done by Coirier, we compared results with a flat plate oriented with the mesh and oriented at an angle to the mesh. The Blasius profiles are shown in Figure 1, and some results for skin friction are shown in Figure 2.

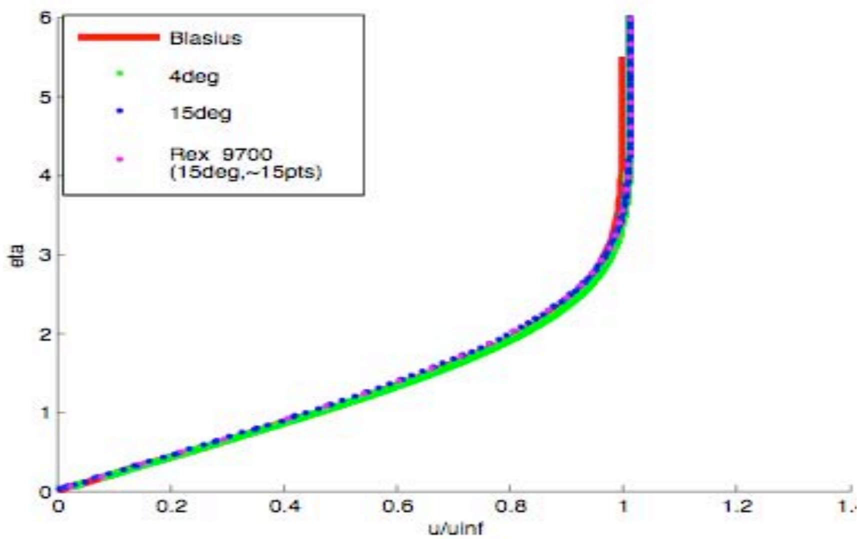

Figure 1 Boundary layer solution in self-similar variables with plate at several angles to the grid, compared to Blasius solution in red.

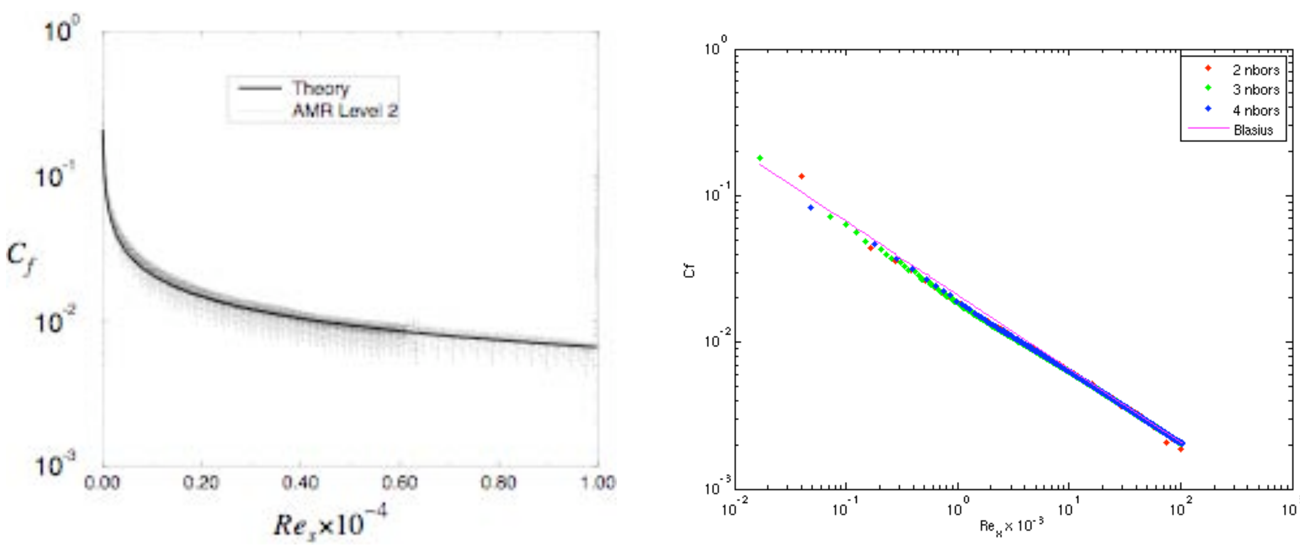

Figure 2 Computed skin friction for the boundary layer compared to the Blasius results. Left, from Coirier's thesis [1]; right, our results (using the more common log-log plot.)

Another test case is flow around a NACA0012 at 0 angle of attack, Re 5000, Mach number .5 The magnitude of the velocity if shown in Figure 3. A plot of Cp and skin friction is shown in Figure 4. The separation point computed was 81.7, well within the range of values computed for this test case.

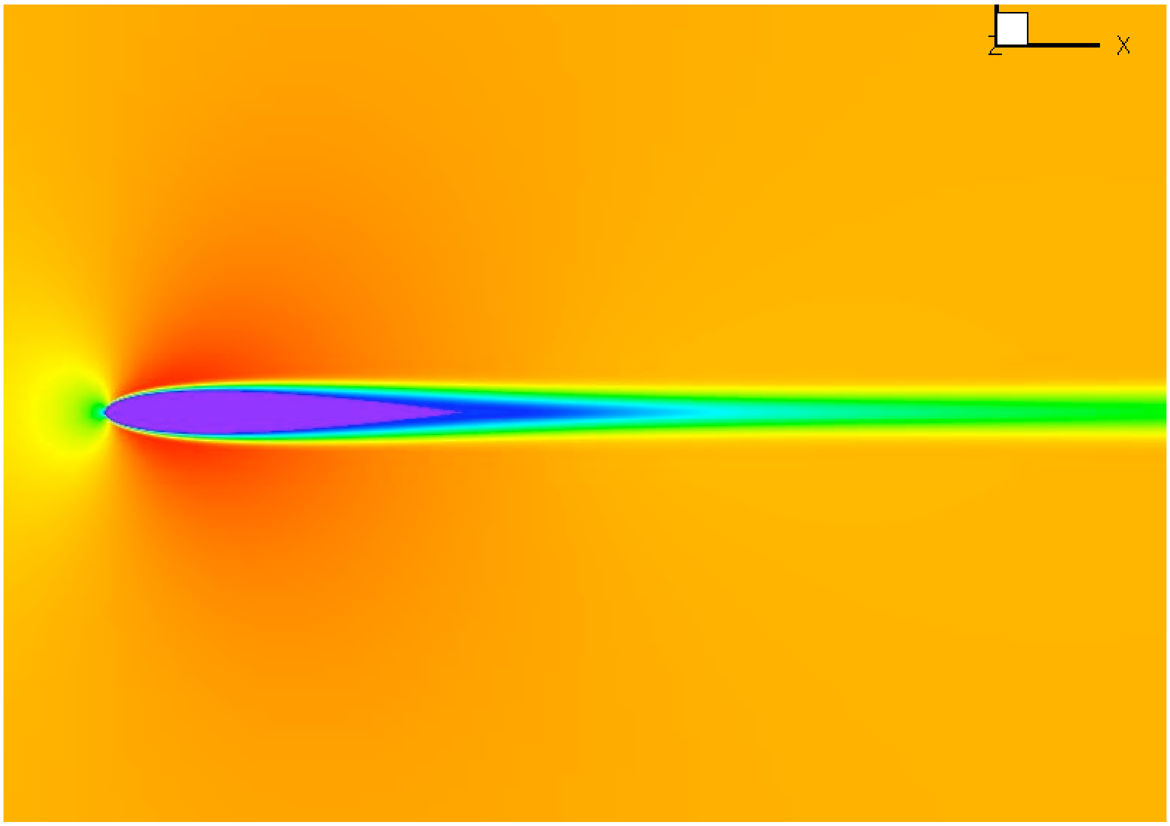

Figure 3 Velocity magnitude is shown around a NACA 0012, with Re 5000, zero angle of attack, and Mach number .5

In figure 4 left we show the pressure coefficient, and on the right the skin, showing the upper and lower surface. When the mesh is not resolved and one can see the results staircasing through the Cartesian grid. Most of this calculation is well resolved, with only a hint of a problem at the leading edge (which is hard to see unless the figures are blown up).

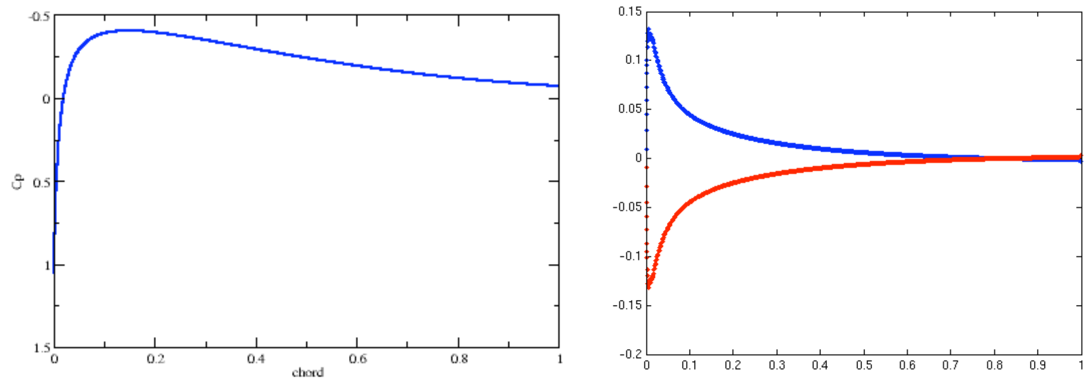

Figure 4 Cp (left) and skin friction (right) for the results on Figure 3 for flow around a NACA 0012.

A revised multigrid acceleration was developed for viscous problems. The theory suggests that higher order derivatives need a better prolongation than the piecewise constant approach we used for inviscid flow. We developed a linear prolongation algorithm, which also seems to help with robustness. To be able to evaluate viscous terms on the coarser grids they need to have some geometry. For example, the coarse grids need to know the cell centroid, surface centroid, cut face information, and other values, to be able to compute gradients on the coarser grids. This involved a substantial amount of work developing the coarse mesh generator. These terms were not needed to compute inviscid flow, since the coarse grids use only a first order method in this case, and no coarser grid gradients are used. Figure 5 shows a convergence plot for the NACA0012 viscous case above. A full multigrid scheme is used, starting on the coarsest level and using five grid levels.

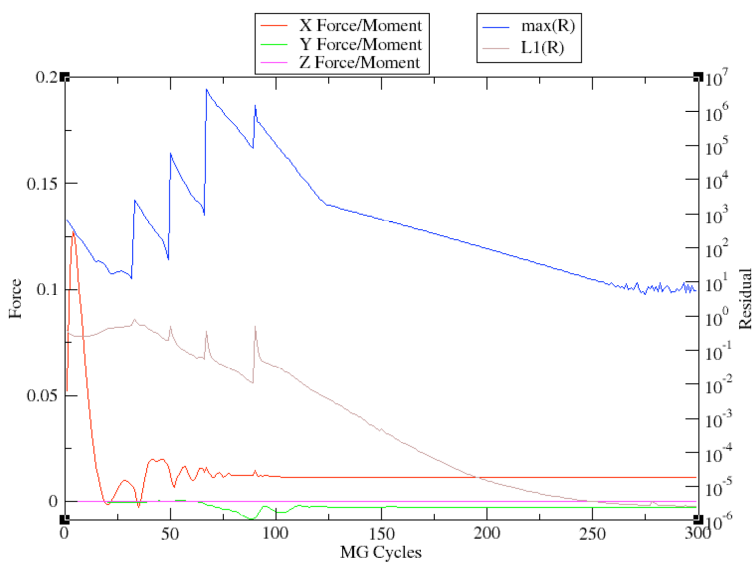

Figure 5 Multigrid convergence for the computation in Figure 3 for flow around a NACA 0012. This case converges in about 200 iterations on the finest grid.

More infrastructure will need to be developed. Our Cartesian mesh generator already has an anisotropic option [2], although it has not been pushed very far or even fully debugged. We developed a new multigrid mesh coarsening algorithm that uses a tree to represent the mesh instead of space-filling curves to generate the coarse meshes. This was necessary since space-filling curves do not retain their nice properties on anisotropically-refined meshes. The tree-based approach will also allow for the option of variable mesh coarsening, for example either semi-coarsening, or more aggressive coarsening when mesh interfaces slow things down. This work was done with my Masters student Greg Tumolo, now graduated. Most important will be the development of a boundary layer model, to reduce the prohibitive cost of isotropic refinement at the boundary. This will be the focus of future research.

References

[1] William Coirier, An Adaptively-Refined, Cartesian, Cell-Cased Scheme for the Euler and Navier-Stokes Equations. PhD Thesis, U. Michigan, 1994.

[2] M. Berger, M. Aftosmis, "Aspects (and Aspect Ratios) of Cartesian Mesh Methods", Lecture Notes in Physics 515, Spring-Verlag. Proc. 16th Intl. Conf. Num. Meth. Fluid Dyn., July, 1998.